Proper Plate Meshing with Mechanical

Introduction

For this discussion, consider the three types of CAD models:

- 3D solid models in which the thickness and size of everything is included in the geometry,
- 3D surface models in which only the outside surface (or inside surface, or the "surface" half way between the inside and outside) is included in the geometry; that is, the thickness of the part is not included in the model,
- or a combination of a solid and surface model.

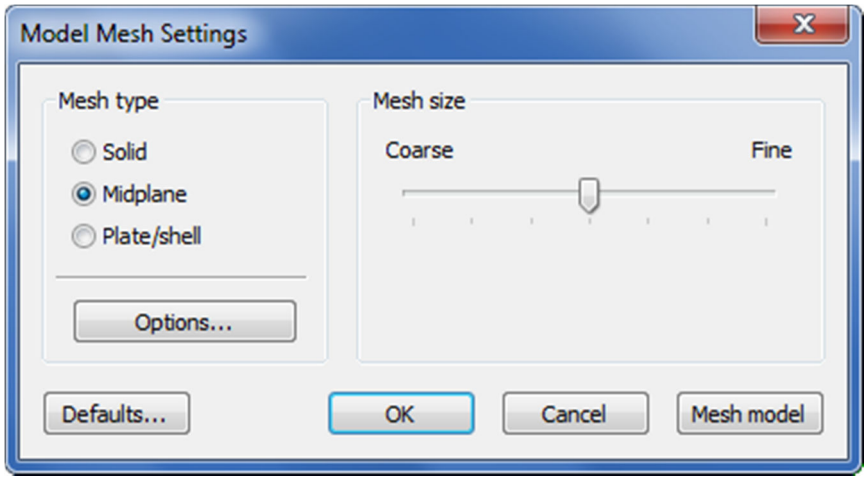
For example, a box or rack to hold electronics may be made as a surface model in the CAD software when the thickness of the material is much smaller than the overall size of the item. But if the thickness is required for whatever reason, this item would need to be made as a solid model.

When meshing a CAD model in Autodesk® Simulation Mechanical, there are three types of meshes that can be selected from the "3D Mesh Settings" dialog (see the figure below):

- Solid mesh in which brick elements fill the volume of the part,
- Midplane mesh in which plate elements are created at the mid-surface between the "top" and "bottom" face of the part, and the original thickness is assigned to the plate elements,
- and Plate/shell mesh which also creates plate elements.

For your information, "plate" element is the name used in linear stress and heat transfer. Shell element is the name used in nonlinear stress. As far as the mesh goes, the two are identical.

This paper discusses which mesh type is appropriate for which CAD model type.



"Mesh > Mesh > 3D Mesh Settings"

The Answer

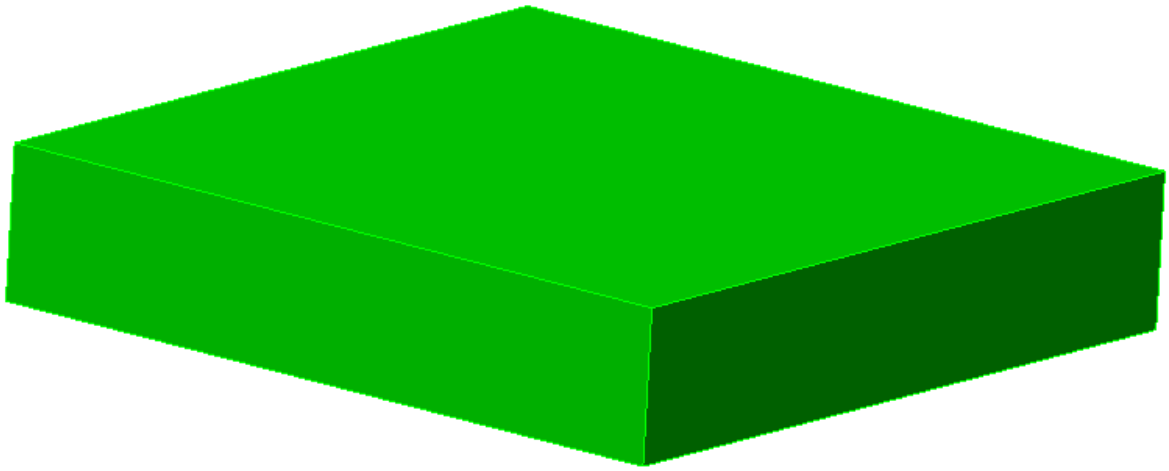
The answer is simple. The following table shows which mesh type can be used with which CAD model type. For example, a solid mesh type can be used when a solid CAD model is imported, but a solid mesh type cannot be used when a surface CAD model is imported. Only the plate/shell mesh type – which creates a mesh with no thickness in 3D space – can be used with a surface CAD model.

Mesh Type	Element Type Created	Type of CAD Model Imported	
		Solid	Surface
Solid	Brick	Acceptable	Not Acceptable
Midplane	Plate or Shell	Acceptable	Not Acceptable
Plate/shell	Plate or Shell	Not Acceptable	Acceptable

The explanation of the answer requires a bit of elaboration.

The Explanation

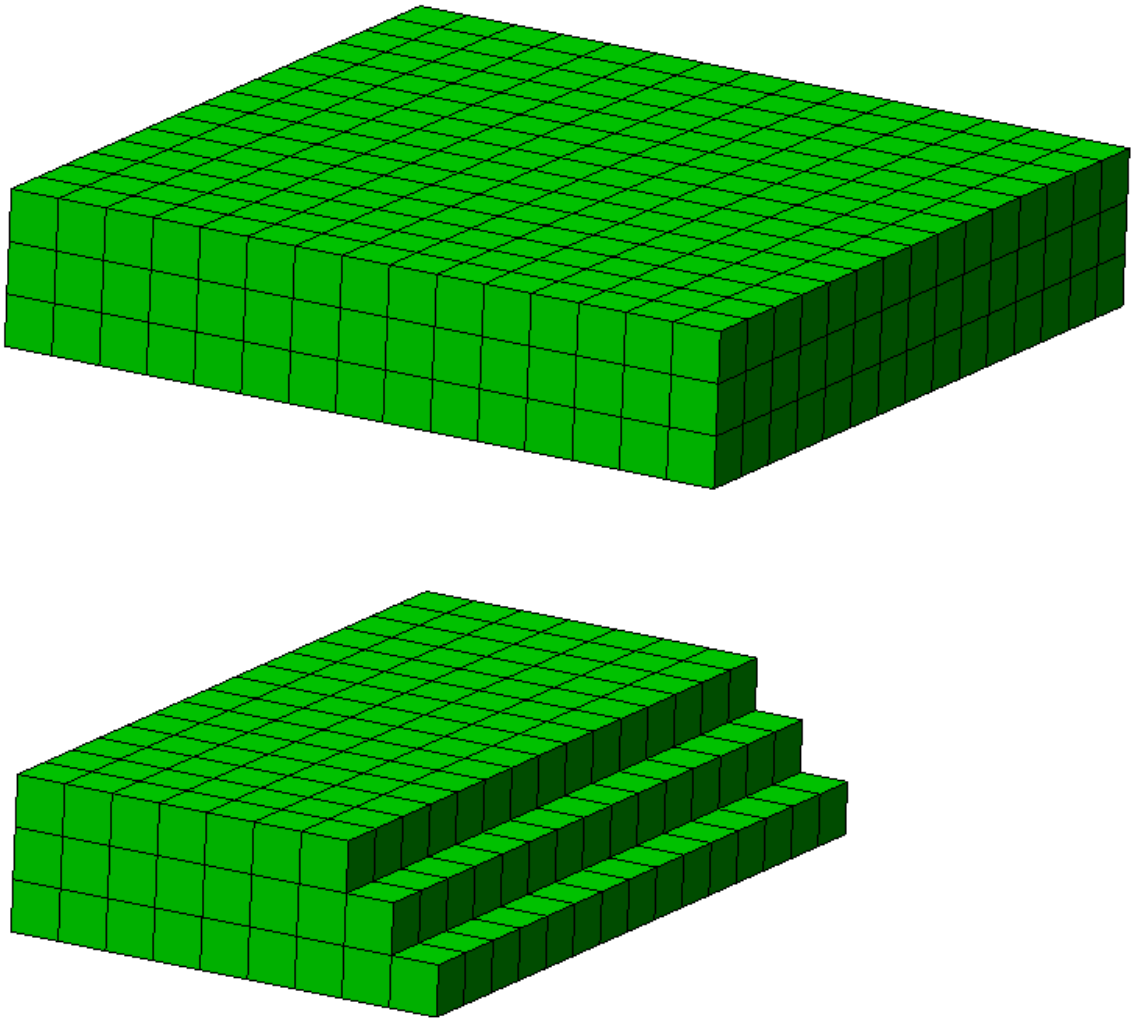
Imagine a simple slab of material with dimensions of 1 unit by 1 unit by 0.2 units. The CAD solid model would look like the following:



CAD Solid Model

Solid Mesh Type

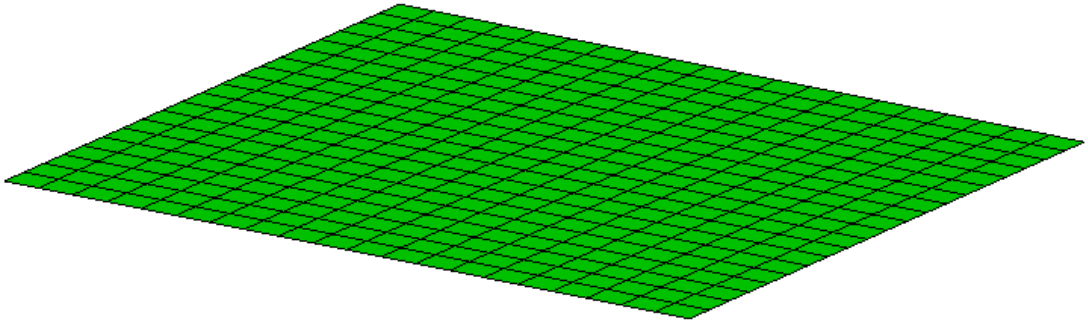
If a solid mesh is created, the volume of the part is filled with brick elements. The completed mesh, showing the full model and with some elements hidden, would look as follows:



A Solid Mesh, showing the complete model (top) and the inside of the part (bottom). Brick elements fill the entire volume. (Images created in the Results environment.)

Midplane Mesh Type

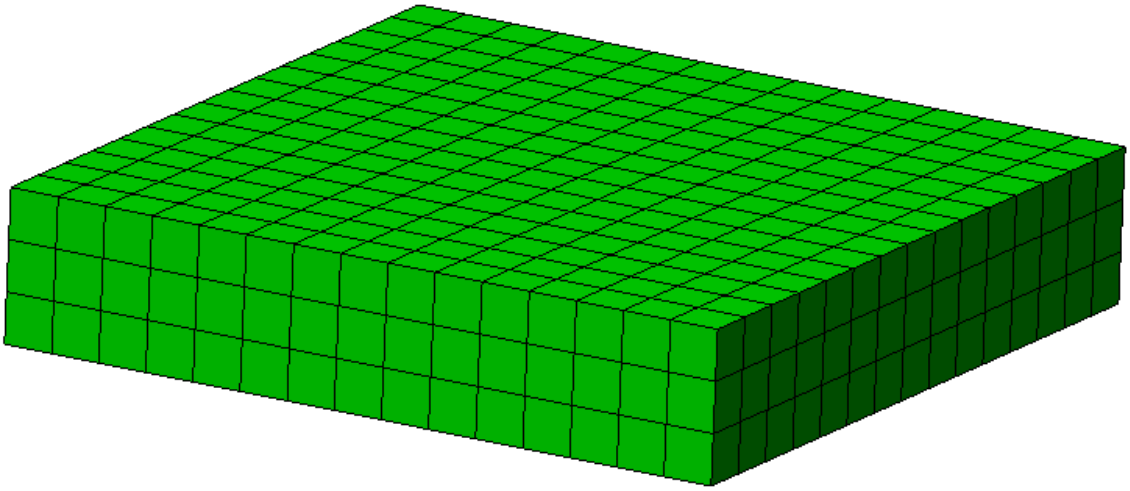
Another way you can think of the midplane mesh type is as follows: collapse the thickness of the solid parts to 0, mesh the resulting surface, and assign a thickness mathematically to each element that represents the thickness of the original part at the location of each element. The resulting mesh looks like a piece of paper as shown in the following figure.



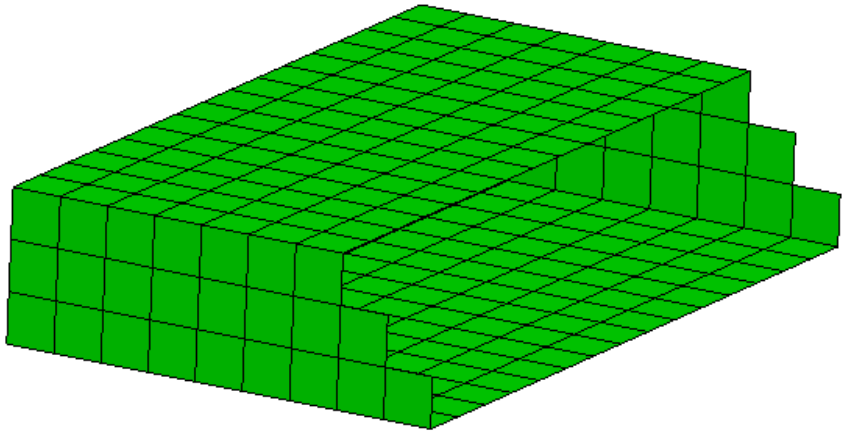
A Midplane Mesh

Plate/Shell Mesh Type

To see why the plate/shell mesh type is not appropriate for CAD solid models, go ahead and mesh the sample model using the plate/shell mesh type. In the Results environment, the complete mesh looks the same as the solid mesh, but hiding elements shows the difference: the surfaces of the "solid" are meshed and made into plate elements (the mesh is 0 thickness, thickness assigned mathematically), but these elements do not represent the volume correctly. If a pressure were applied to the top face of the part, it would bend as two separate plates (held together by plates around the perimeter) instead of bending as one plate with the proper thickness.

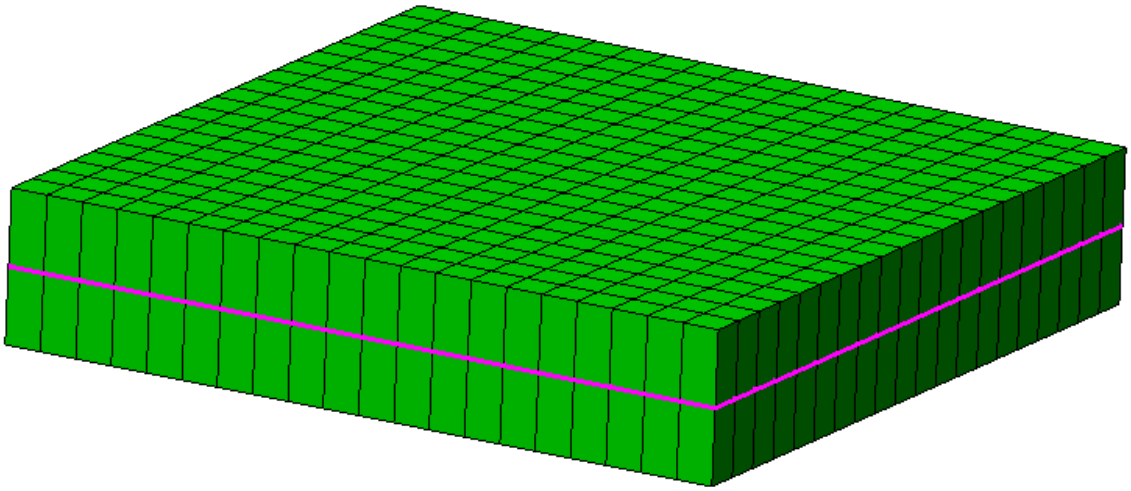


A Plate/Shell Mesh. This view looks like the original CAD solid, so what is wrong?

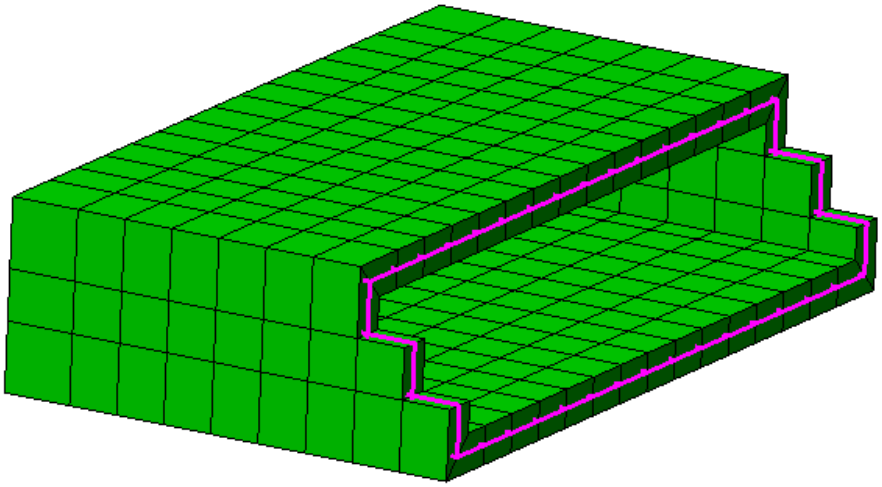


The Same Plate/Shell Mesh With Some Elements Hidden. The slab has been turned into a hollow box. There is no way that this model will accurately represent the real part.

The thickness of plate elements can be shown in the Results environment. Right-click on a part that uses plate elements in the browser and activate “3-D Visualization”. The two plate models shown previously are shown below with the 3-D visualization turned on.



The Midplane Mesh With 3-D Visualization. The pink line shows the location of the plate elements. Half of the assigned thickness is above the plate element, and half of the thickness is below the plate element. This model looks like the original slab and is acceptable.



The Plate/Shell Mesh With 3-D Visualization. The pink line shows the location of the plate elements. This model does not look like the original slab, so it is not acceptable. (The thickness has been set to 0.05 units to better illustrate the "hollow box" that the mesh represents.)

History

Revision	Date	Author	Description of Change
0	2013 March 5	John Holtz (AstroJohnPE)	Original document using Simulation Mechanical 2013.